VISIT…

TABLE OF CONTENTS

*Release Notes* ..... ix
**1. INTRODUCTION** ..... 1
1. TYPES OF ANALYSIS ..... 1
1. DC Analysis ..... 1
2. AC Small-Signal Analysis ..... 2
3. Transient Analysis ..... 2
4. Pole-Zero Analysis ..... 2
5. Small-Signal Distortion Analysis ..... 2
6. Sensitivity Analysis ..... 3
7. Noise Analysis ..... 3
2. ANALYSIS AT DIFFERENT TEMPERATURES ..... 4
3. CONVERGENCE ..... 6
**2. CIRCUIT DESCRIPTION** ..... 7
1. GENERAL STRUCTURE AND CONVENTIONS ..... 7
2. TITLE LINE, COMMENT LINES AND .END LINE ..... 8
1. Title Line ..... 8
2. .END Line ..... 8
3. Comments ..... 8
3. DEVICE MODELS ..... 9
4. SUBCIRCUITS ..... 10
1. .SUBCKT Line ..... 10
2. .ENDS Line ..... 10
3. Subcircuit Calls ..... 11
5. COMBINING FILES: .INCLUDE LINES ..... 12
**3. CIRCUIT ELEMENTS AND MODELS** ..... 13
1. ELEMENTARY DEVICES ..... 13
1. Resistors ..... 13
2. Semiconductor Resistors ..... 13
3. Semiconductor Resistor Model (R) ..... 14
4. Capacitors ..... 15
5. Semiconductor Capacitors ..... 15
6. Semiconductor Capacitor Model (C) ..... 16
7. Inductors ..... 16
8. Coupled (Mutual) Inductors ..... 17
9. Switches ..... 17
10. Switch Model (SW/CSW) ..... 18
2. VOLTAGE AND CURRENT SOURCES ..... 19
1. Independent Sources ..... 19
1. Pulse ..... 20

2. Sinusoidal ..... 21
3. Exponential ..... 21
4. Piece-Wise Linear ..... 22
5. Single-Frequency FM ..... 23
2. Linear Dependent Sources ..... 23
1. Linear Voltage-Controlled Current Sources ..... 24
2. Linear Voltage-Controlled Voltage Sources ..... 24
3. Linear Current-Controlled Current Sources ..... 24
4. Linear Current-Controlled Voltage Sources ..... 25
3. Non-linear Dependent Sources ..... 25
3. TRANSMISSION LINES ..... 27
1. Lossless Transmission Lines ..... 27
2. Lossy Transmission Lines ..... 27
3. Lossy Transmission Line Model (LTRA) ..... 28
4. Uniform Distributed RC Lines (Lossy) ..... 29
5. Uniform Distributed RC Model (URC) ..... 30
4. TRANSISTORS AND DIODES ..... 31
1. Junction Diodes ..... 31
2. Diode Model (D) ..... 32
3. Bipolar Junction Transistors (BJTs) ..... 33
4. BJT Models (NPN/PNP) ..... 33
5. Junction Field-Effect Transistors (JFETs) ..... 35
6. JFET Models (NJF/PJF) ..... 36
7. MOSFETs ..... 37
8. MOSFET Models (NMOS/PMOS) ..... 38
9. MESFETs ..... 43
10. MESFET Models (NMF/PMF) ..... 43
**4. ANALYSES AND OUTPUT CONTROL** ..... 45
1. SIMULATOR VARIABLES (.OPTIONS) ..... 45
2. INITIAL CONDITIONS ..... 49
1. .NODESET: Specify Initial Node Voltage Guesses ..... 49
2. .IC: Set Initial Conditions ..... 49
3. ANALYSES ..... 50
1. .AC: Small-Signal AC Analysis ..... 50
2. .DC: DC Transfer Function ..... 50
3. .DISTO: Distortion Analysis ..... 51
4. .NOISE: Noise Analysis ..... 52
5. .OP: Operating Point Analysis ..... 53
6. .PZ: Pole-Zero Analysis ..... 53
7. .SENS: DC or Small-Signal AC Sensitivity Analysis ..... 54
8. .TF: Transfer Function Analysis ..... 54
9. .TRAN: Transient Analysis ..... 55

4. BATCH OUTPUT ..... 56
1. .SAVE Lines ..... 56
2. .PRINT Lines ..... 56
3. .PLOT Lines ..... 57
4. .FOUR: Fourier Analysis of Transient Analysis Output ..... 58
**5. INTERACTIVE INTERPRETER** ..... 59
1. EXPRESSIONS, FUNCTIONS, AND CONSTANTS ..... 60
2. COMMAND INTERPRETATION ..... 64
3. COMMANDS ..... 65
1. Ac*: Perform an AC, small-signal frequency response analysis ..... 65
2. Alias: Create an alias for a command ..... 65
3. Alter*: Change a device or model parameter ..... 65
4. Asciiplot: Plot values using old-style character plots ..... 66
5. Aspice: Asynchronous spice run ..... 66
6. Bug: Mail a bug report ..... 66
7. Cd: Change directory ..... 66
8. Destroy: Delete a data set ..... 67
9. Dc*: Perform a DC-sweep analysis ..... 67
10. Define: Define a function ..... 67
11. Delete*: Remove a trace or breakpoint ..... 68
12. Diff: Compare vectors ..... 68
13. Display: List known vectors and types ..... 68
14. Echo: Print text ..... 68
15. Edit*: Edit the current circuit ..... 69
16. Fourier: Perform a fourier transform ..... 69
17. Hardcopy: Save a plot to a file for printing ..... 69
18. Help: Print summaries of Spice3 commands ..... 70
19. History: Review previous commands ..... 70
20. Iplot*: Incremental plot ..... 70
21. Jobs: List active asynchronous spice runs ..... 70
22. Let: Assign a value to a vector ..... 70
23. Linearize*: Interpolate to a linear scale ..... 71
24. Listing*: Print a listing of the current circuit ..... 71
25. Load: Load rawfile data ..... 71
26. Op*: Perform an operating point analysis ..... 72
27. Plot: Plot values on the display ..... 72
28. Print: Print values ..... 72
29. Quit: Leave Spice3 or Nutmeg ..... 73
30. Rehash: Reset internal hash tables ..... 73
31. Reset*: Reset an analysis ..... 73
32. Reshape: Alter the dimensionality or dimensions of a vector ..... 74
33. Resume*: Continue a simulation after a stop ..... 74

34. Rspice: Remote spice submission ..... 74
35. Run*: Run analysis from the input file ..... 75
36. Rusage: Resource usage ..... 76
37. Save*: Save a set of outputs ..... 77
38. Sens*: Run a sensitivity analysis ..... 77
39. Set: Set the value of a variable ..... 77
40. Setcirc*: Change the current circuit ..... 78
41. Setplot: Switch the current set of vectors ..... 78
42. Settype: Set the type of a vector ..... 78
43. Shell: Call the command interpreter ..... 78
44. Shift: Alter a list variable ..... 79
45. Show*: List device state ..... 79
46. Showmod*: List model parameter values ..... 80
47. Source: Read a Spice3 input file ..... 80
48. Status*: Display breakpoint information ..... 80
49. Step*: Run a fixed number of timepoints ..... 81
50. Stop*: Set a breakpoint ..... 81
51. Tf*: Run a Transfer Function analysis ..... 81
52. Trace*: Trace nodes ..... 82
53. Tran*: Perform a transient analysis ..... 82
54. Transpose: Swap the elements in a multi-dimensional data set ..... 82
55. Unalias: Retract an alias ..... 83
56. Undefine: Retract a definition ..... 83
57. Unset: Clear a variable ..... 83
58. Version: Print the version of Spice ..... 83
59. Where: Identify troublesome node or device ..... 84
60. Write: Write data to a file ..... 84
61. Xgraph: use the xgraph(1) program for plotting. ..... 84

4. CONTROL STRUCTURES ..... 85
1. While — End ..... 85
2. Repeat — End ..... 85
3. Dowhile — End ..... 85
4. Foreach — End ..... 86
5. If — Then — Else ..... 86
6. Label ..... 86
7. Goto ..... 87
8. Continue ..... 87
9. Break ..... 87

5. VARIABLES ..... 88

6. MISCELLANEOUS ..... 92

7. BUGS ..... 94

**6. BIBLIOGRAPHY** ..... 97

**A. APPENDIX A: EXAMPLE CIRCUITS** ..... 99
1. Circuit 1: Differential Pair ..... 99
2. Circuit 2: MOSFET Characterization ..... 99
3. Circuit 3: RTL Inverter ..... 100
4. Circuit 4: Four-Bit Binary Adder ..... 100
5. Circuit 5: Transmission-Line Inverter ..... 102
**B. APPENDIX B: MODEL AND DEVICE PARAMETERS** ..... 103
1. URC: Uniform R.C. line ..... 104
2. ASRC: Arbitrary Source ..... 104
3. BJT: Bipolar Junction Transistor ..... 105
4. BSIM1: Berkeley Short Channel IGFET Model ..... 108
5. BSIM2: Berkeley Short Channel IGFET Model ..... 110
6. Capacitor: Fixed capacitor ..... 114
7. CCCS: Current controlled current source ..... 115
8. CCVS: Linear current controlled current source ..... 115
9. CSwitch: Current controlled ideal switch ..... 115
10. Diode: Junction Diode model ..... 116
11. Inductor: Inductors ..... 117
12. mutual: Mutual inductors ..... 118
13. Isource: Independent current source ..... 118
14. JFET: Junction Field effect transistor ..... 119
15. LTRA: Lossy transmission line ..... 120
16. MES: GaAs MESFET model ..... 121
17. Mos1: Level 1 MOSfet model with Meyer capacitance model ..... 123
18. Mos2: Level 2 MOSfet model with Meyer capacitance model ..... 126
19. Mos3: Level 3 MOSfet model with Meyer capacitance model ..... 129
20. Mos6: Level 6 MOSfet model with Meyer capacitance model ..... 132
21. Resistor: Simple linear resistor ..... 135
22. Switch: Ideal voltage controlled switch ..... 135
23. Tranline: Lossless transmission line ..... 136
24. VCCS: Voltage controlled current source ..... 137
25. VCVS: Voltage controlled voltage source ..... 137
26. Vsource: Independent voltage source ..... 138

## RELEASE NOTES

Spice3f is the six major release of Spice3. This release incorporates new features not available in Spice 3c or 3d, as well as several performance improvements. All of the features described here are believed to be fully functional. The development of SPICE and its algorithms is ongoing at Berkeley, and therefore not all of the intended capabilities of the program have been implemented in full yet.

**Bugs in 3f2 fixed in 3f3**:

```
Ascii (printer) plots in spice3f2 did not print bode plots
vs log of the frequency by default, as in spice2.
You had to explicity request the x-axis to be log; either
"plot vdb(2) xlog" (best) or "plot vdb(2) vs log10(frequency)" will do.
Now simply "plot vdb(2)" will work.

The on-line documentation has been brought up to date by converting this
manual into a format readable on-line.

Significant problems with AC sensitivities in 3f2 only have been fixed.

Multiple analyses and plots in spice2 emulation mode under 3f2 and earlier
generated misleading error messages.
This no longer happens in 3f3.
```

**New features and bug fixes incorporated in Spice3f** (the current release of Spice3):

```
Sensitivity analysis.
Added a parameter for fitting JFET models (parameter "B").
Fixed a discontinuity problem in MOS level 3
      (related to the "kappa" parameter).
Working "alter" command.
Improved "show" and "showmod" commands for operating
      point summary tables (like Spice2).
Working "trace" command.
Interactive "set" variable values now the same as ".options" settings.
Improved plotting, including implicitly transforming data for smith plots.
Added the function "deriv( )" (derivative) to the front-end.
Corrected an error affecting the specified initial conditions for
      some circuits.
Small signal behaviour of non-linear dependant sources fixed.
Miscellaneous bug fixes in the front end.
```

**New features and bug fixes incorporated in Spice3e** (the *previous* release of Spice3):

```
Lossy Transmission Lines.
Proper calculation of sheet resistance in the MOS models.
A new command ("where") to aid in debugging troublesome circuits.
Smith-chart plots working (see the "plot" command).
Arbitrary sources in subcircuits handled correctly.
Arbitrary source reciprocal calculations and DC biasing fixed.
Minor bug-fixes to the Pole-Zero analysis.
Miscellaneous bug fixes in the front end.
```

**Some common problems remaining in Spice3f** (note that this list is not complete):

```
Models defined within subcircuits are not always handled correctly.
If you have trouble, move the model definition outside of ".subckt"
and ".ends" lines.
```

```
Batch run data is not compacted if a "rawspice" data file is generated,
resulting in excessively large output files for some difficult inputs.

Sufficient detail is sometimes not preserved in transient analysis.
Providing a small value for the "TMAX" parameter (the fourth argument)
in the transient run command will solve this problem.

Convergence problems can sometimes be worked around by relaxing the
transient "TMAX" parameter.

The substrate node of the bipolar transistor (BJT) is modelled
incorrectly (this may actually be due to inherant numerical
problems with the model).  Do not use the substrate node; use
a semiconductor capacitor to model substrate effects.

Charge is not conserved in MOS devices based on the Meyer model.

Transient simulation of strictly resistive circuits (typical for
first runs or tests) allow a time step that is too large (e.g. a
sinusoidal source driving a resistor).  There is no integration error
to restrict the time step.  Use the "TMAX" parameter or include reactive
elements.

Deep nesting of subcircuits may exceed internal static buffers.

The PZ analysis can not be interrupted;  the sensitivity analysis
can not be continued (the interactive "resume" command) once interrupted.

There are many other small bugs, particularly in the front end.
```

**Bug reporting:**

Please send bug reports or suggested enhancements to Spice3 via electronic mail to:

```
spice@Berkeley.EDU  (Internet)
ucbvax!spice  (UUCP-net)
```

or by physical mail to "Spice3 Bug Report" at:

```
EECS/ERL Industrial Support Office
205 Cory Hall
U.C. Berkeley
Berkeley, CA   94720
```

Information on ordering Spice3 or other software distributed through the EECS/ERL Industrial Support Office should be directed to the above physical mail address or to "ilpsoftware@berkeley.edu" via electronic mail.

Include the version of Spice used, the system type and operating system version, the smallest input file demonstrating the problem, and the resulting (incorrect) output.